1.0 INTRODUCTION

PROGRAMME NAME	: LABFIL
VERSIONS	: BOTH EXE, BAS
PURPOSE	: Linear Thrust Analysis of any section under axial and bending moments
USABLE ON SYSTEM	: IBM COMPATIBLE PC XT
LATEST ON DATE	: 24.10.91

2.0 TYPE OF STRUCTURE WHICH CAN BE ANALYSED

The programme can analyse a section of any shape with any number of voids under axial thrust and bending moments. It can be a plain or RCC or partially prestressed concrete section or steel section or a foundation resting on soil.

This programme is written in BASIC language acceptable for IBM Compatible PC. XT.

The programme can be used for linear analysis of any section under axial thrust and bending moments. The section (and voids, if any) is described with the help of nodes along the boundary of the section. The reinforcement is described in a similar manner. The programme plots the total section and reinforcement and gives the sectional properties. Under the axial thrust and bending moments specified, the programme carries out a number of iterations each time changing the values of position of N.A. and the maximum stress until they satisfactorily balance the imposed loads. The programme also plots the cracked section. With the final values of position of N.A. and maximum stress , the programme can also give the strain and stresses at any point of the section and the cracked section properties.

The programme prepares Input and Output files which can be either displayed on the screen or printed with the help of a Printer.

4.0 ASSUMPTIONS MADE

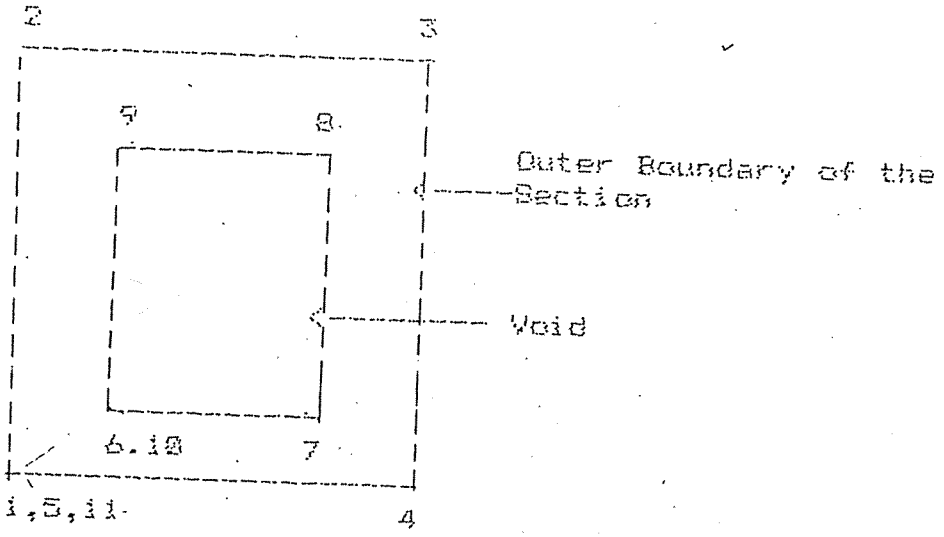
The programme is meant for Linear Analysis only.

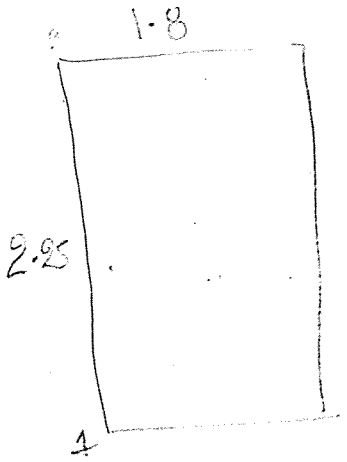
The orientation of the imposed loads and the positions of prestressing cables if any are to be arranged in such a manner that the node nearest to the origin of the global axis system is under maximum compression.

5.0 DISCRETIZATION TO BE FOLLOWED

The programme can analyse a section of any shape with any number of voids. Start numbering the nodes from any node. Number other nodes successively moving in a clockwise direction ending in the 1st node and numbering it again.

If there is a void in the section then after having numbered the outer boundary as given above, move to the node nearest to the first node and number it next. Now moving in the anti-clockwise direction, number the nodes successively along the boundary of the void. Ending it on to the 1st node of the void, number it again. From there come back to the first node of the section and number it the third time. Proceed in a similar fashion if you have more number of voids in the section. Refer the figure given below for the number of sequence.





For circular section, a different approach is followed. The coordinates of the node at which the circle or arc of a circle starts and the coordinates of the centre of the circle are described. The angle Theta subtended by the arc at the centre of the circle (for a complete circular section this angle would be 360 deg) and the number of segments in which you would like to divide the arc is input. The programme will do the numbering of the next nodes internally depending on the angle and the number of segments.

It should be noted that in case of circular arcs, the nodes generated by the computer are joined by straight line segments. Therefore, smoother arch can be achieved by defining more no. of segments.

NUMBERING FOR CIRCULAR SECTION

Circular voids are described in a similar fashion except that the angle subtended by the circle or arc of a circle at the centre is described negative to indicate the anti-clockwise numbering sequence for voids.

For reinforced sections, if you have a number of bars of the same diameter spaced uniformly along a straight line or an arc of a circle, you can describe them together as "Continuous" reinforcement. If you have no such uniformity in the reinforcement, then each bar is to be described.

For describing the global coordinate system, draw the X-axis touching the bottom-most node of the section. X and Y coordinates in this system are then input for all the nodes of the section and reinforcement.

IMPORTANT : P and Mx, My should be defined in such a way that max. compression occurs at node no. 1 and max tension occurs at bar no. 1. Otherwise, it will give absurd / wrong results. (i.e. P-downward, Mx +ve and My -ve, if the 1st node is on the left bottom corner and 1st reinf. bar is at right top corner).

6.0 INSTRUCTION FOR USE

1. Create an Input file using EDLIN or any suitable operation, assigning it a suitable name and extension.
2. Run the Programme file LABFIL of desired version (i.e. EXE or BAS) with the prescribed procedure.
3. Programme will ask for Input file name which can be entered.
4. Programme displays the uncracked shape of the structure including the bars if any. This can be printed using Pri Sc (Print Screen).
5. Programme gives the options to the user if he desires to get cracked section properties, stress and strain at extra points. Programme can be commanded as per requirements.
6. In case user opts for the cracked section properties, the programme works out by itrative procedure location of cracked portion and displays its properties (eg Area, intercepts of the line of the common line between cracked and uncracked portions with X and Y axes. This diagram can also be printed using Pri Sc.
7. At the end of the programme user can PRINT the following files :

LAB1.OUT : File of Input Data created by the programme in a prescribed format. This filp also contains the sign convention for user friendliness.

LAB2.OUT : This is an output file which contains the following -
a) Properties of Gross Section
b) Information mentioned in 5 and 6 above only if opted for during the execution of the programme.

NOTE : It should be noted that everytime the programme LABFIL is run, the above two files get updated by the input and output respectively of the new problem solved. Therefore, in case the user wants to retain the contents of the previously existing files LAB1.OUT and LAB2.OUT then he can do so by copying them in a different file name prior to running the programme.

7.0 CREATING AN INPUT FILE

An Input file can be created using any suitable operation on PC and assigning it a suitable name and extension. As a normal practice the extension can be DAT to represent a data file. Following are the elements of an Input File with their corresponding explanations :

EXPLANATION :

A

B C

D E F G H

I J K L

Mh Nh Oh

P Q R

Si Ti Ui Vi Wi Xi Yi

Z

Ai Bi Ci

A : A suitable title for one's own information
B : Material Type - 1 or 2 or 3
    1. Soil
    2. Concrete
    3. Steel
C : E value of the Material defined in B above.

E : Shrinkage coeff., Creep Factor of Material only in case of Concrete. If the element B is 1 or 3 no value should be assigned here.

F : Allowable tensile stress in the material

G : No. of nodes on outer face.

H : Total no. of nodes

I : No. of Passive Bars, Prestressing bars only in case of Concrete. if the element B is 1 or 3 , no value should be assigned here.

K : E value of Passive Steel. Only if I is non-zero. No value should be assigned if element I is zero.

L : E value of Prestressing Steel : Only if J is non-zero. No value should be assigned if element J is zero.

Nh : X and Y coordinates of H number of nodes. Datum for coordinates should be chosen in such a way that no coordinate is -ve even in case of circular arc in which case they are calculated by the programme. Node numbering should be done in such a way that node no. 1 attracts the max compressive stress. (Node nos. are automatically assumed as 1,2,3...They should not be specified separately).

Oh : Angle in degrees to define circular arc. This value should be zero in case of discrete or straight line member (-ve in case of void defining internal circle).

P : No. of segments in circular arc. It should be noted that if number of segments is given any value say P then coordinates of P+1 number of nodes are assumed to be automatically defined which are calculated by the programme itself. (See Note 1 below).

Q, R : X and Y coordinates of centre in case of an arc. (See Note 1 below)

NOTE 1 : a) Elements P,Q,R should appear only if element Oh is non zero. Otherwise no value should be assigned.

b) It should be noted that programme assumes the circular arc defined by user as consisting of straight line segments equal to the number of segemnts being defined. Therefore to achieve a finer circular arc, it is required to feed more number of segments (generally 16 to 32 nos) and accordingly the number of nodes.
c) It should be noted that once P+i number of nodes have been defined, the further node number to be defined will be previous node number +(P+i).

Si, Ti : X, Y coordinates of 1st reinforcing bar. (See Note 2 below). Numbering should be done in such a way that bar no. 1 attracts the maximum Tensile Force.

Ui, Vi : X, Y coordinates of last reinforcing bar in case of straight line and centre of circle in case of circular arc.
Vi = Si, Ui = Ti, in case of discrete bars
(See Note 2 below)

Wi : Area of bar / bars

Xi : No. of bars (at least 1)

Yi : Angle in degrees subtended by circular arc. Zero for straight line. 360 deg for complete circle.

NOTE 2: a) Elements Si,Ti,Ui,Vi,Wi,Xi,Yi,Zi shall occur only if element I is non zero. Otherwise no value should be assigned.

DESIGN OF PIER P2R SEC A (SARDAND) TOP OF FOOTING

MAX - M35 (A)

2 2.5E 6 — B, C

0 0 0 34 67 — D, E, F, G, H

216 0 2E7 — I, J, K, L (Reinforcing bar, prestressing bar, E)

2.7 0 360 → Mx, Ny (Ox - Ry)

32 2.7 2.7 → P, Q, R [No. of segments, Centre of arc]

2.7 0.5 -360

32 2.7 2.7

2.7 0 0

2.7 5.332 2.7 2.7 8.69E-2 108 356.66 — R/F IN OUTER RING - 108-Φ32

2.7 4.968 2.7 2.7 8.69E-2 108 356.66 — R/F IN INNER RING - 108-Φ32

(1st bar coord, centre coord, Area of bar, no. of bars, Angle)

1.NORMAL ONE SPAN LOADED

2008 3886 0

2.NORMAL TWO SPANS LOADED

2190.2 3942 0

3.VERT AND LONGL SEISMIC Pmax

2381.9 8966 0

4.VERT AND LONGL SEISMIC Pmin

1998.5 8966 0

5.VERT AND TRANS SEISMIC ONE SPAN LOADED Pmax

2216 7533 0

6.VERT AND TRANS SEISMIC ONE SPAN LOADED Pmin

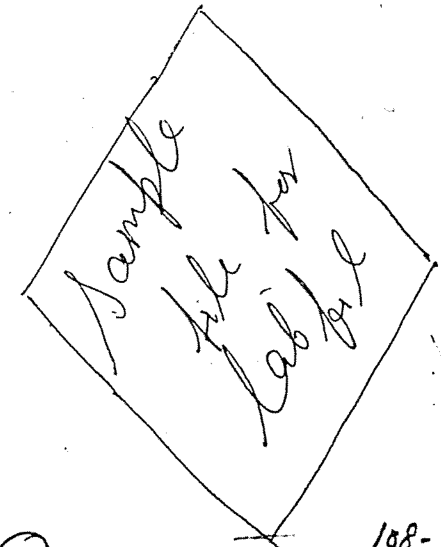
1800.5 7533 0

7.VERT AND TRANS SEISMIC TWO SPANS LOADED Pmax

2410 8541.3 0

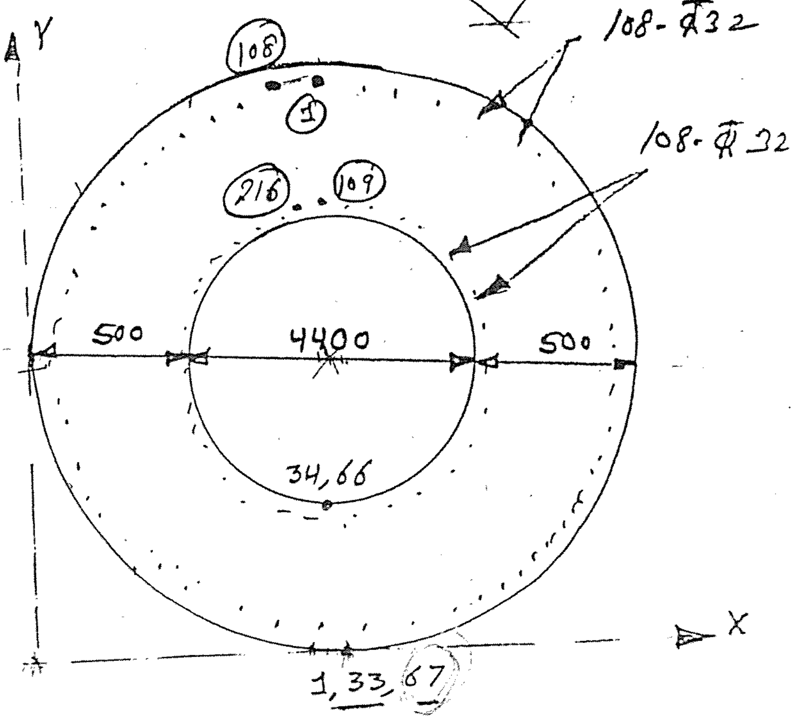
8.VERT AND TRANS SEISMIC TWO SPANS LOADED Pmin

1970.5 8541.3 0



Pier Node No. - 1 to 67

R/F Node No. - 1 to 216



Bar 1 attracts maximum ...

Node 1 attracts maximum ...

b) It should be noted that coordinates and location of bars are independent of the geometry of section being defined. Therefore user should make sure that reinforcing / prestressing bars do not fall outside the section.

Z : Title for load case for user's own information. (Computer will itself print load cases 1,2,etc for load cases).

NOTE 3 : It is necessary to INPUT some value of Z,A,B & C even to get sec. properties alone

A1, Vertical load (+ve downward).

B1, Bending moment about global X-axis.

C1 : Bending moment about Y axis, (+ve directions by right hand screw rule.

Direction of loads and moments and orientation of structure should be so chosen as to produce maximum compression at node 1 and maximum tension at bar no. 1.